



THE ANALYSIS OF FLUID DYNAMICS OF WAVE POWER STATION WITH WELLS TURBIN BY CFD

Kim-Bang Tran, Quang-Sang Nguyen, Tich-Thien Truong*

Department of Engineering Mechanics, Faculty of Applied Science, Ho Chi Minh city University of Technology, VNU-HCMC, 268 Ly Thuong Kiet Street, District 10, Ho Chi Minh City

*Email: tttruong@hcmut.edu.vn

Received: 22 July 2019; Accepted for publication: 13 January 2020

Abstract. Natural energy such as wind, wave and other natural vibrations is one of the high potential renewable energy sources. The Wells turbine is based on the use of bidirectional turbines, which act as axial-flow self-rectifying turbines that employs a symmetrical blade profile and rotating unidirectionally in reciprocating airflows generated by the air chamber to extract energy from vibrations. These topics have been extensively studied both numerically and experimentally such as research on the parameters of the effects of structure, angle of attack, blade shape, etc. In this paper, numerical simulation is carried out using commercially available tool Fluent for fluid dynamics analysis and focus on oscillating predictions, with particular attention to the behavior of the flow. Based on the Numerical Wave Tank (NWT) model is simulated in a two dimensional used in this model, which is constructed mainly based on the spatially averaged Navier Stokes equation with the k- ϵ model for simulating the turbulence and modelled with Volume of Fluid (VOF). Axial-flow turbines system and future development as well as the proposed limitations will be discussed in detail.

Keywords: Oscillating Water Column (OWC), Volume of Fluid (VOF), dynamic flow, wave energy, Computational fluid dynamics (CFD).

Classification numbers: 3.4.1, 5.4.4.

1. INTRODUCTION

Wave energy converter (WEC) is considered a promising renewable energy source. They have the ability to harness the periodic rise and fall of the wave. Oscillating Water column (OWC) is one of the most important wave energy conversion device of the most developed WEC types. The OWC uses the air flow produced by the pressure change inside the oscillating water column, which is open below the water surface, allowing water to flow in and out an internal chamber and generating electricity. Multiple authors have conducted to better understand the performance of different OWC devices, the fundamentals to perform mathematical modeling of OWC devices [1]. Studies on OWC's properties and performance based on linear wave theory and nonlinear small amplitude waves theory are conducted [2, 3]. The instantaneous water surface elevation inside the OWC chamber is spatially variant and can only be regarded as uniform when the chamber size meets the design requirements that the

studies have given [4, 5]. the product of air pressure in the chamber and the air flow rate of an OWC using a two phase VOF model in FLUENT, and found that the amplitudes of the wave elevation within the chamber obtained from the 2D Numerical Wave Tank (NWT) compared well with experimental data [6]. Because of this bidirectional air flow, axial-flow Wells turbines are used in most prototypes. It was introduced in 1976 at Queen's University in Belfast some research into different aerodynamic blade profiles of the Wells turbine and modifications related to the airflow velocities [7-9]. A series of useful studies on OWC have been carried out by some authors from the application of the theory and boundary conditions to improve the properties and performance of OWC, the numerical model the flow was assumed to be inviscid, unsteady and incompressible were modeled in a NWT two-dimensional and k- ϵ turbulent model, where the Volume of Fluid (VOF) model were used [10-17]. ANSYS Fluent is a commercial software which solves the continuity and momentum Navier-Stokes equations based on the finite volume method. The aim of the present study is to carry out a numerical study to calculate the effect of the model on the wave amplitude relative to the direction of the flow of the chamber front wall.

2. NUMERICAL SETUP

2.1. Geometric model

The Oscillating Water Column devices are, basically, steel or concrete hollow structures partially submerged in a rectangle with a rectangular vent above.

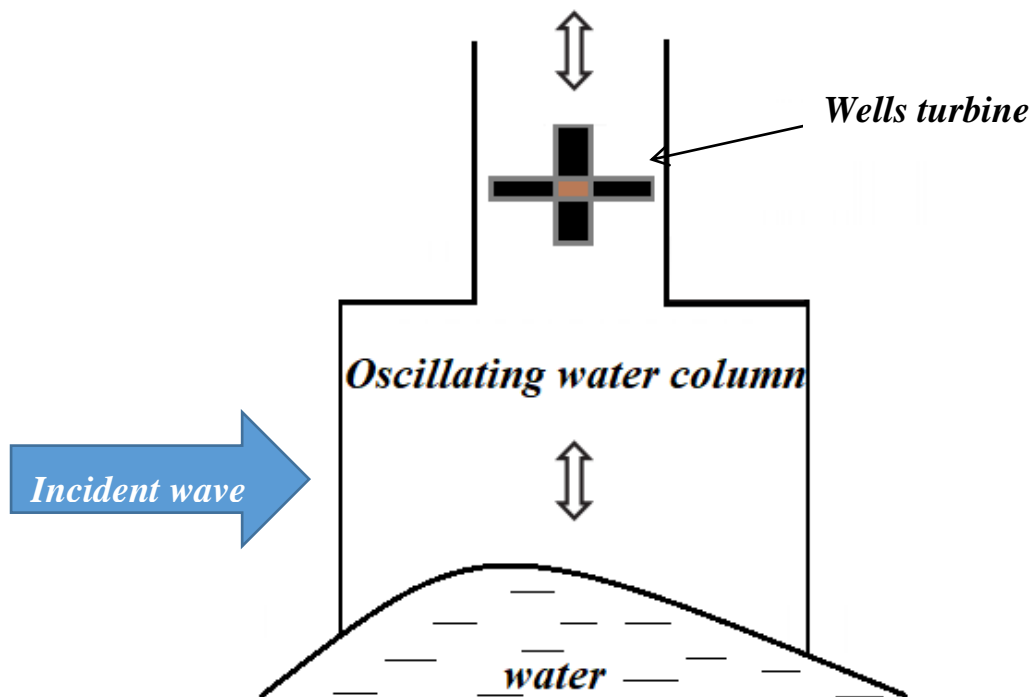


Figure 1. Schematic diagram of oscillating water column.

The NWT dimension has been introduced in Figure 2 shows a schematic and the boundary condition of NWT which were implemented. Reflection appearance is an important issue that

should be considered through wave simulation. In fact, the reflection occurs when the ocean waves strike a solid object and is negatively affected such as the impact on the cliffs or in the NWT experiments or in digital wave tank experiments, the waves will be bounced off when exposed to the wall. In physical modeling, wave energy will be dissipated using porous material in the end of the wave flume. In numerical approach there is also several methods for example using coarser mesh or using a longer flume. Another way is to use a porous zone. In this paper, a longer flume is considered to investigate the wave amplitude for a period of time how long OWC's performance will change.

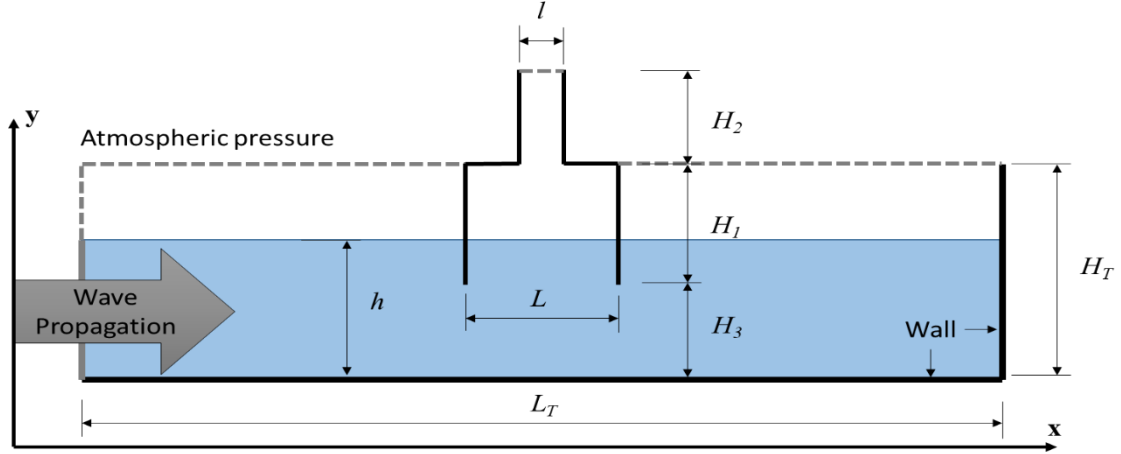


Figure 2. Schematic representation of the computational domain .

The regular and in real scale wave in the present work has the following characteristics, where water depth is $h = 10.0$ m, wave height is $H = 1.0$ m, the period T is set to 5s, $L_T = 188$ m and $H_T = 13$ m. Besides, as already mentioned, the hydropneumatic chamber length $L = 16.7097$ m and height $H_1 = 2.2501$ m, chimney outlet length $l = 2.3176$ m and its height $H_2 = 6.9529$ m, and the submergence depth $H_3 = 9.5$ m of the device were defined in agreement with the best OWC shape proposed by [12], being these values given by: $L = 16.7097$ m, $H_1 = 2.2501$ m, $l = 2.3176$ m, $H_2 = 6.9529$ m and $H_3 = 9.5$ m.

2.2. Mathematical model

The continuity and momentum (Navier-Stokes) equations based on the finite volume method. This is the dominant equation that the fluid problem is based on and it is integrated into Fluent tools in ANSYS:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \quad (i=1,2) \quad (1)$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \rho f_i + F_i \quad (i=1,2) \quad (2)$$

where: ρ is the specific mass of the fluid density (kg/m^3), t is the time (s), p is the static pressure (N/m^2), μ is the molecular viscosity (kg/m.s), $\bar{\tau}$ is the stress tensor (N/m^2) and \bar{g} is the gravitational acceleration (m/s^2); x_i , x_j are longitudinal coordinate and vertical coordinate

respectively, u_i (m/s) represent horizontal and vertical velocity components respectively and $f_1 = 0$, $f_2 = -g$, F_i are the source terms.

In this study, the VOF method is applied for consequently free surface capture is an indispensable part of this study. Hence, other governing equations are introduced. The VOF formulation is based on the fact that two or more phases are immiscible. This condition is used to delineate the limits of two phases without penetrating each other. In each control volume, the sum of the volume fraction of all phases:

$$\sum_{q=1}^n f_q = 1 \quad (3)$$

In each volume, if $f_q = 1$ shows that the volume is full of water, when $f_q = 0$ the volume contain only air and if $0 < f_q < 1$ the volume shows the interface.

The geometrical characteristics of the device are shown in Figure 2. the wave maker is placed in the left side of the wave tank. The wave characteristics are chosen so that it is well represented by Stokes second order solution. This methodology consists of applying the horizontal u and vertical v components of wave velocity as boundary conditions (velocity inlet) of the computational model, by means a User Defined Function (UDF) by Fluent tools in the ANSYS software [18]. The wave considered in this study has a constant period T , wavelength λ and incoming wave height H . According to the second order theory, the equation of the free surface elevation is [19]:

$$\eta = \frac{H}{2} \cos(kx - \omega t) + \frac{H^2}{16} \frac{k \cosh(kh)}{\sinh^4(kh)} (2 + \cosh 2kh) \cos 2(kx - \omega t) \quad (4)$$

$$u = \frac{\partial \phi}{\partial x} = \frac{Hgk}{2\omega} \frac{\cosh k(y+h)}{\cosh(kh)} \cos(kx - \omega t) + \frac{3H^2 \omega k}{16} \frac{\cosh 2k(h+y)}{\sinh^4(kh)} \cos 2(kx - \omega t) \quad (5)$$

$$v = \frac{\partial \phi}{\partial y} = \frac{Hgk}{2\omega} \frac{\cosh k(y+h)}{\cosh(kh)} \sin(kx - \omega t) + \frac{3H^2 \omega k}{16} \frac{\cosh 2k(h+y)}{\sinh^4(kh)} \sin 2(kx - \omega t) \quad (6)$$

In Equations (5, 6) the first function is linear, the second function is Stoke 2nd.

No slip wall boundary conditions were set at the bottom wall, back wall and chamber wall. The top outlet and of the numerical tank were set as constant pressure boundaries. The velocity and free surface elevation of the water for the inlet at the left hand end of the domain were defined so as to represent an incident wave according equations (4-6). Vertical elements were variable also, so that finer mesh was applied near still water level where the variation of water level and capturing free surface level was important.

The standard k- ϵ turbulence model has been used to simulate mean flow characteristics for turbulent flow conditions

$$\rho \frac{Dk}{dt} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \epsilon \quad (7)$$

$$\rho \frac{D\epsilon}{dt} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + \frac{\epsilon}{k} (C_{\epsilon 1} P_k - C_{\epsilon 2} \rho \epsilon) \quad (8)$$

where, the characteristic parameters $\sigma_k = 1$, $\sigma_\epsilon = 1.3$, $C_{\epsilon 1} = 1.44$, $C_{\epsilon 2} = 1.92$, $C_\mu = 0.09$. It is often used for the problem with relatively small pressure gradients that fit the model in this paper.

In this article, geo-reconstruct scheme in Figure 3 is enabled to capture the free surface. This scheme is the most accurate and is applicable for general unstructured meshes. The first step in this reconstruction scheme is to calculate the position of the linear interface relative to the center of each partially-filled cell, based on information about the volume fraction and its derivatives in the cell. The second step is to calculate the advection amount of fluid through each face using the computed linear interface representation and information about the normal and tangential velocity distributions on the face. The third step is to calculate the volume fraction in each cell using the balance of fluxes calculated during the previous step.

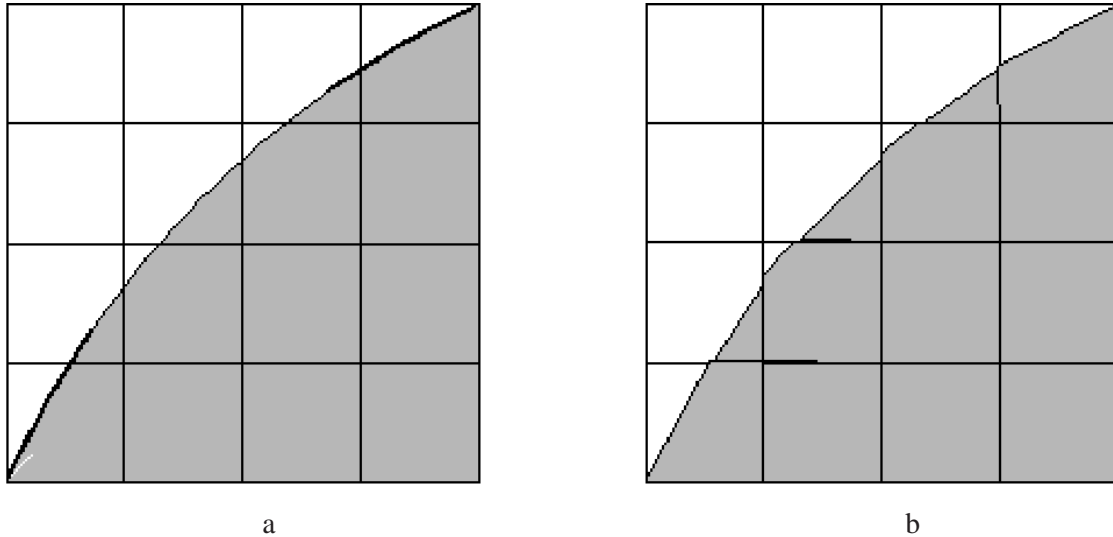


Figure 3. (a) The curve boundary of a phase and (b) using geo-constructed scheme.

3. SIMULATION RESULTS

The differences between the analytical and numerical results arise from the assumptions for the mathematical modelling in both approaches. To summarize, in the analytical approach, it's assumed that the air flow is compressible, the air chamber remains stationary and Bernoulli's theorem is used for the air flow through the nozzle. Thus, the problem is considered one dimensional and the wave profile outside the OWC is assumed to be sinusoidal. In the numerical approach, VOF Model feature of the Fluent software is used, so that the flow is governed by incompressible Navier Stokes equations where the flow is viscous, and the wave profile is generated by the wall motion of the numerical wave tank. As a result, two dimensional velocity and pressure fields are obtained for both water and air phases. Calculations in this study are performed by considering an incident wave that physical parameters satisfy the Stokes second-order theory. In all transient analysis performed in the present study the time step is set to 0.01 s. The movement of the free surface is obtained by converting the hydrostatic pressure to a height of fluid column. The numerical solution and analytical solution for wave amplitude are listed in Figures 4-6 and simulation field in ANSYS software in Figure 7 and Figure 8 as follows:

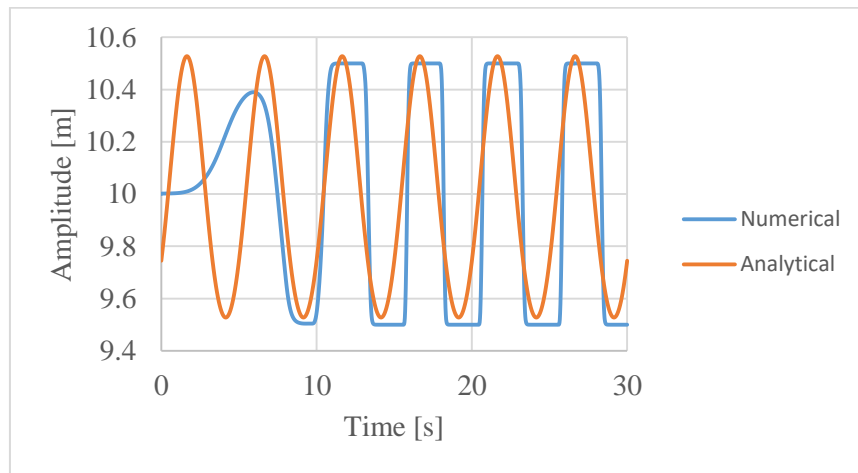


Figure 4. Wave amplitude at $x = 50$ m.

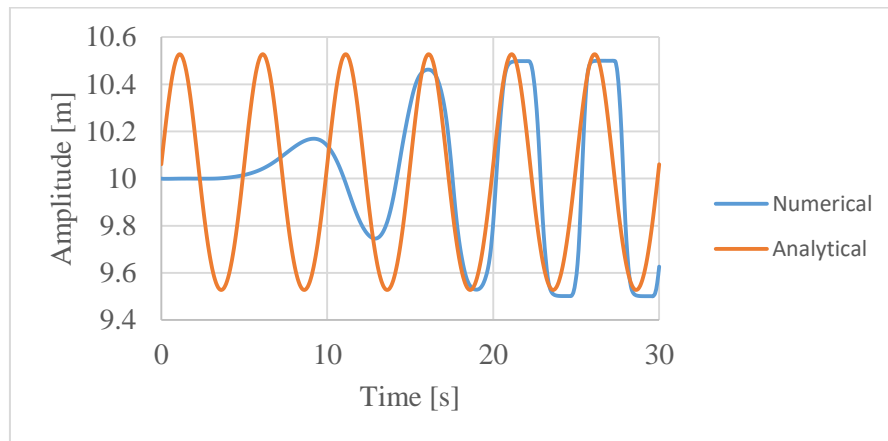


Figure 5. Wave amplitude at $x = 83.56$ m.

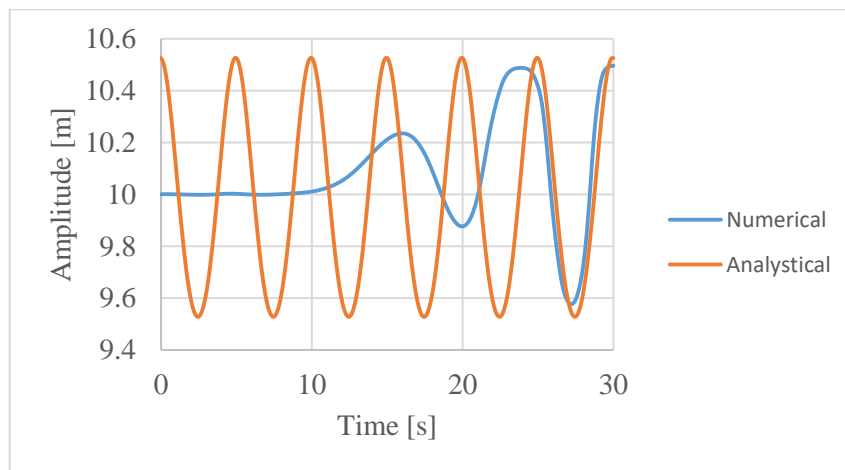


Figure 6. Wave amplitude at $x = 150$ m.

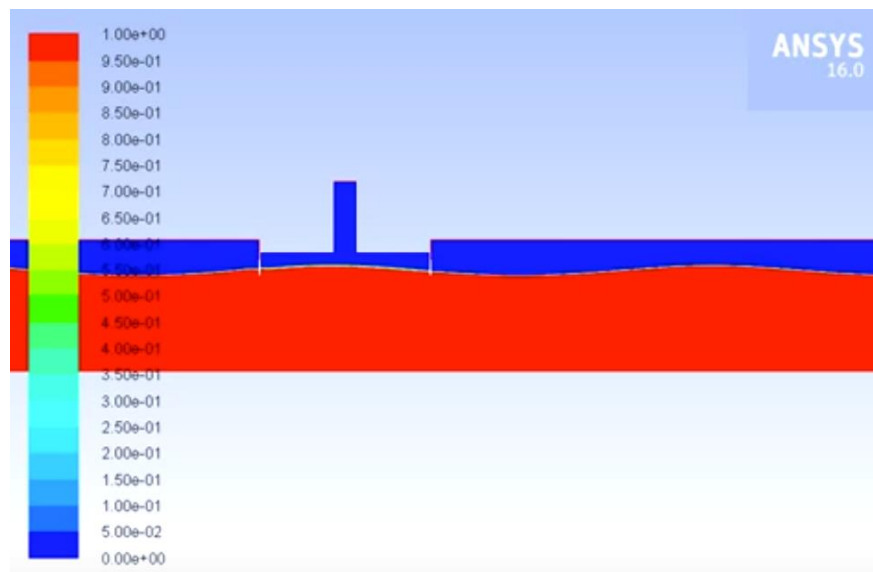


Figure 7. Wave propagation.

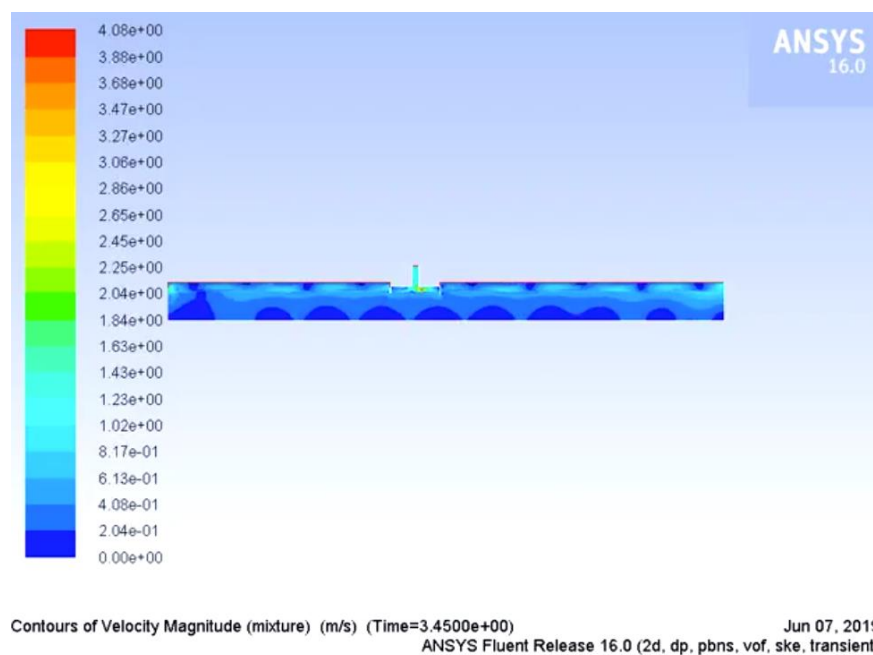


Figure 8. Velocity field.

4. CONCLUSION

The simulation results show that the wave velocity and amplitude are suitable for oscillating water column device in this paper, which uses the air flow produced by the pressure change inside the column. The numerical results of wave amplitude show that at the maximum and minimum position of the wave or wave crest position, the shape tend to maintain state which

is difference with the shape of the wave crest of the analytical results due to the solution based on steady solution while the transient solution will show the true shape of the wave crest but usually consuming computer cost.

The analytical results are based on the theory of a compressible air flow neglecting the losses in the fluid-fluid and the fluid-structure interactions while flow was assumed to be viscous and incompressible for the numerical model. Possibility of financial support to obtain the equipment for the wave tank experiments reveals the third approach, the experimental method, which will be the next subject of research of this study.

REFERENCE

1. Sarmento A. J. N. A. and Falcao A. F. de O. - Wave generation by an oscillating surface pressure and its application in wave energy extraction, *Journal of Fluid Mechanics* **150** (1984) 467-485.
2. Evans D. V. - The oscillating water column wave energy device, *Journal of the Institute of Mathematics and its Applications* **22** (1978) 423-433.
3. Ursell E., Dean R. G., and Yu Y. S. - Forced Small Amplitude Water Waves: A Comparison of Theory and Experiment, *J. Fluid Mech.* **7** (1960) Pt. 1.
4. Evans D. V. and Porter R. - Hydrodynamic Characteristics of an Oscillating Water Column Wave Energy Device, *Applied Ocean Research* **17** (1995) 155-164.
5. Evans D. V. and Porter R. - Efficient Calculation of Hydrodynamic Properties of OWC-type devices, *J. Offshore Mech. Arctic Eng.* **119** (1997) 210-218.
6. Zhang Y., Zou Q. P., Deborah G. - Air water two-phase flow modeling of hydrodynamic performance of an oscillating water column device. *Renewable Energy* **41** (2012) 159-170.
7. Torresi M., Camporeale S. M., Strippoli P. D., Pascazio G. - Accurate numerical simulation of a high solidity Wells turbine, *Renewable Energy* **33** (4) (2008) 735-747.
8. Kim T., Takao M., Setoguchi T., Kaneko K., and Inoue M. - Performance comparison of turbines for wave power conversion, *International Journal of Thermal Sciences* **40** (2001) 681-689.
9. Setoguchi T., Santhakumar S., Takao M., Kim T. H., and Kaneko K - A modified Wells turbine for wave energy conversion, *Renewable Energy* **28** (2003) 79-91.
10. Gomes M. das N., Olinto C. R., Rocha L. A. O., Souza J. A., and Isoldi L. A. - Computational modeling of a regular wave tank, *Engenharia Térmica* **8** (2009) 44-50.
11. Gomes M. das N., Nascimento C. D., Bonafini B. L., Dos Santos E. D., Isoldi L. A., and Rocha L. A. O. - Two-dimensional geometric optimization of an oscillating water column converter in laboratory scale, *Engenharia Térmica* **11** (2012) 30-36.
12. Gomes M. das N., Dos Santos E. D., Isoldi L. A., and Rocha L. A. O. - Two-dimensional geometric optimization of an oscillating water column converter of real scale. In: 22nd International Congress of Mechanical Engineering (COBEM), Ibero Latin American Congress on Computational Methods in Engineering (CILAMCE) 2013, pp. 1-12.
13. Horko M. - CFD Optimization of an Oscillating Water Column Energy Converter, Master Thesis, Science and Engineering, School of Mechanical Engineering, University of Western, Australian, 2007.

14. Liu Z., Hyun B., and Hong K. - Application of Numerical Wave Tank to OWC Air Chamber for Wave Energy Conversion, in: International Offshore and Polar Engineering Conference, Vancouver, BC, Canada, 2008, pp. 350-356.
15. Liu Z., Hyun B., and Jin J. - Numerical Prediction for Overtopping Performance of OWEC, Journal of the Korean Society for Marine Environmental Engineering **11** (1) (2008) 35-41.
16. Liu Z., Hyun B., and Hong K. - Numerical Study of Air Chamber for Oscillating Water Column Wave Energy Converter, China Ocean Eng. **25** (2011) 169-178.
17. Lopes N. R., Sant'Anna F. S. P., Gomes M. N., Souza J. A., Teixeira P. R. F., Dos Santos E. D., Isoldi L. A., and Rocha L. A. O. - Constructal Design Optimization of the Geometry of an Oscillating Water Column Wave Energy Converter (OWC-WEC), in: Proceedings Constructal Law Conference, UFRGS, Porto Alegre, 2012.
18. FLUENT, 2017 - "User's Manual". ANSYS, Inc.
19. Dean R. G., Dalrymple R. A. - Water Wave Mechanics for Engineers and Scientists, World Scientific Publishing Company, 1991.